

***Bounding Estimates For The
'Hot' Channel Coolant
Temperature And Preliminary
Calculation Of Mixing In The
Lower Plenum For The NGNP
Point Design Using CFD***

Next Generation Nuclear Plant (NGNP)

Richard W. Johnson

Richard R. Schultz

December 2004



*Idaho National Engineering and Environmental Laboratory
Bechtel BWXT Idaho, LLC*

Bounding Estimate For The 'Hot' Channel Coolant Temperature And Preliminary Calculation Of Mixing In The Lower Plenum For The NGNP Point Design Using CFD

Richard W. Johnson
Richard R. Schultz

December 2004

Idaho National Engineering and Environmental Laboratory

Idaho Falls, Idaho 83415

Prepared for the
U.S. Department of Energy
Assistant Secretary for Nuclear Energy
Under DOE Idaho Operations Office
Contract DE-AC07-99ID13727

EXECUTIVE SUMMARY

One of the concerns regarding the Next Generation Nuclear Plant (NGNP) is whether the materials will be subjected to temperatures that will challenge the plant's structural integrity. Hence it is important to investigate the factors that will influence maximum coolant temperatures in the core and the trajectory of the coolant jets that may impinge on the lower plenum structures including the mixing between adjacent coolant jets. The work described herein gives preliminary calculations of the maximum coolant temperature at the inlet to the lower plenum and of the interactive mixing of the coolant jets as they move into and through the lower plenum.

Calculation of Maximum Coolant Temperature at Hot Channel Exit: The power density in the core of the block next generation nuclear power plant (NGNP) will not be uniform due to geometry, core layout and fuel pin design. Recent calculations performed to optimize the core design indicate that the maximum radial variation will be 1.25 times the average. This significant radial variation in the local power density will create a variation in the temperature of the helium coolant as it cools the core. The coolant channel with the highest outlet temperature is referred to as the 'hot' channel. The concern is that the coolant in the high temperature channels, which exits into the lower plenum as jets, called 'hot streaking,' will adversely affect materials in the lower plenum, the exit duct and the downstream power conversion equipment. The objective of the present study is to determine or bound the maximum exit temperature of the 'hot' channel. The maximum hot channel temperature depends on the total coolant flow rate, which has not yet been fixed.

The helium in the hot channel will be hotter not only because the local power density is higher, but also because the flow will slow down due to higher wall friction of the hotter fluid. Hence, the increase in temperature of the hot channel is compounded by these two effects. Also, because the lower flow rate in the hot channel cannot be accurately estimated using a simple heat balance, it must be estimated using a fine grid numerical solution which will track continuous changes in fluid properties that affect the flow rate. A commercial computational fluid dynamics (CFD) code called Fluent is used to make these calculations. To ensure the accuracy of the CFD computations, guidelines published by the *ASME Journal of Fluids Engineering* are followed. The critical aspect of the flow computations is the wall friction which directly impacts the fluid flow rate. It is shown that a particular 'enhanced wall treatment' used in conjunction with the standard $k\sim\epsilon$ turbulence model produces accurate results for the wall friction, validating the computations.

Figure i illustrates the outlet temperatures of the hot and average coolant channel computed as a function of total core flow rate assuming that the flow rate is the same in both the hot (red line) and average (blue line) channel. Actually, the flow rate in the hot channel is less than in the corresponding average channel. Three realistic hot channel

outlet temperatures are shown (symbols) for three cases computed with Fluent. These cases have core pressure drops of 4, 5 and 6 psi. The bypass flow is assumed to be 15%.

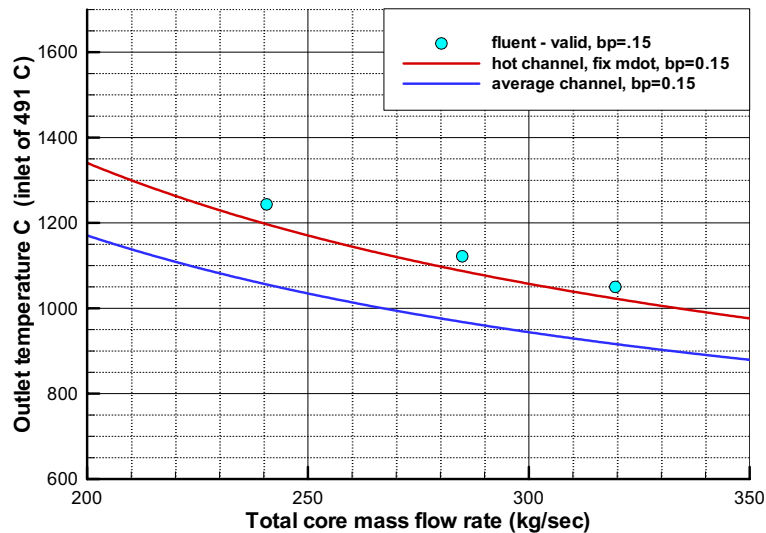


Figure i. The outlet temperatures of the average and hot channel assuming the same flow rates plus the realistic Fluent hot channel results (symbols). The Fluent cases are for pressure drops of 4, 5 and 6 psi (left to right).

Preliminary Calculation of Mixing in Lower Plenum: The fluid dynamics in the lower plenum of the NGNP involves the downward vertical flow of scores of hot turbulent jets of coolant exiting the core and entering into the lower plenum amid the crossflow of the coolant as it turns 90 degrees and heads to the single exit duct while negotiating the dozens of cylindrical core support columns. It is desired to be able to numerically simulate the complex flow and heat transfer in the lower plenum to address concerns about materials and uniformity of temperature at the turbine inlet. It is fully expected that a sophisticated computational fluid dynamics (CFD) code will be used. However, due to the complexity of the flow, it will be necessary to validate the CFD code, particularly the turbulence model or turbulence modeling strategy.

Experiments need to be designed to capture the complex physics of the lower-plenum flow to allow assessment and validation of numerical simulations. While preliminary CFD simulations are not yet validated, they can be of use in the planning of the experiments, particularly in estimating where there are regions of high and low turbulence intensity. Mixing of the coolant is related to the turbulence intensity as well as to the overall nature of the mean flow. The purpose of the present task is to provide preliminary flow calculations of the coolant in the lower plenum to examine flow patterns and turbulence intensity. Figure ii shows contours of turbulence intensity on a horizontal plane at the midpoint of the lower plenum. Because the lower plenum is symmetrical about the centerline of the hot duct that delivers the helium to the power conversion equipment, Figure ii only shows half of the plenum. The white circles represent the solid

support columns which have no flow. The colored field gives a measure of the turbulence intensity. The flow is from right to left where the helium exits the lower plenum via the hot duct. The calculation shows the turbulence intensity to progressively increase from right to left.

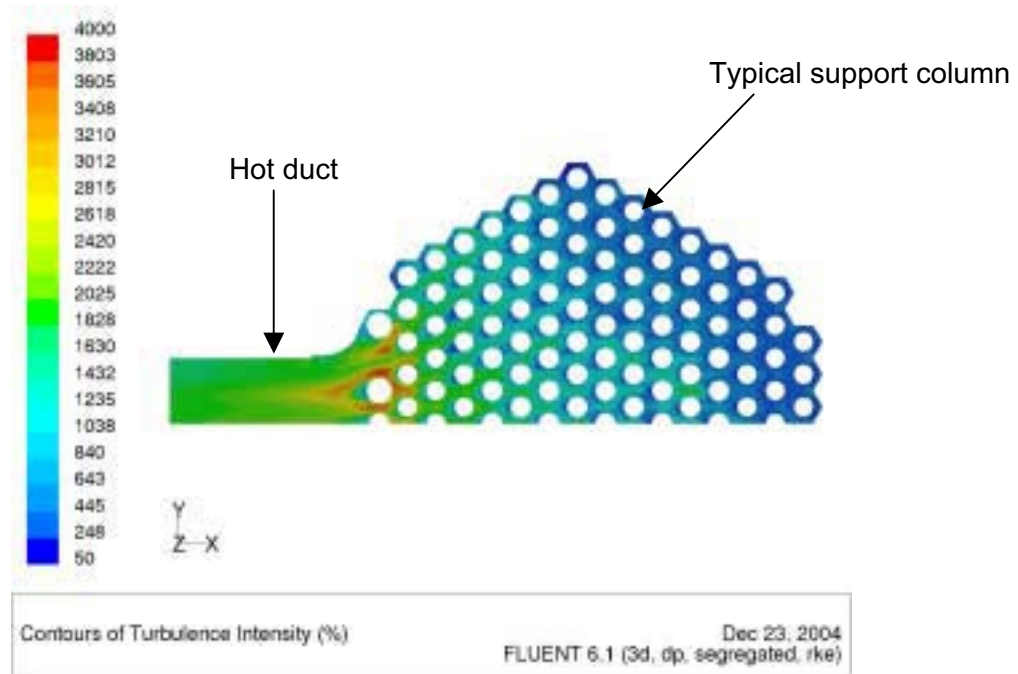


Figure ii. Contours of turbulence intensity on a horizontal plane at the midpoint of the lower plenum.

TABLE OF CONTENTS

EXECUTIVE SUMMARY	V
1 INTRODUCTION	1
2 DESCRIPTION OF HOT CHANNEL COOLANT TEMPERATURE CALCULATIONS	1
2.1 APPROACH	3
2.2 NUMERICAL CONSIDERATIONS.....	9
2.3 RESULTS AND DISCUSSION.....	12
3 MIXING IN THE LOWER PLENUM	18
3.1 OBJECTIVES.....	19
3.2 APPROACH AND RESULTS.....	19
4 CONCLUSIONS	25
5 ACRONYMS.....	25
6 REFERENCES.....	27

1 INTRODUCTION

One of the concerns regarding the Next Generation Nuclear Plant (NGNP) is whether the materials will be subjected to temperatures that will challenge the plant's structural integrity. Hence it is important to investigate the factors that will influence maximum coolant temperatures in the core and the trajectory of the coolant jets that may impinge on the lower plenum structures including the mixing between adjacent coolant jets. The work described herein gives preliminary calculations of the maximum coolant temperature at the inlet to the lower plenum and of the interactive mixing of the coolant jets as they move into and through the lower plenum.

The hot channel coolant temperature calculations are summarized in Section 2. The preliminary lower plenum mixing calculations are summarized in Section 3. Conclusions regarding both of these topics are given in Section 4.

2 DESCRIPTION OF HOT CHANNEL COOLANT TEMPERATURE CALCULATIONS

The power density in the core of the block NGNP will not be uniform due to the geometry and layout of the core and the design of the fuel compacts, even though efforts will have been made to minimize the variation. Figure 1 illustrates the layout of the core of the block (prismatic) NGNP point design. Recent calculations performed during core design optimization efforts indicate that the maximum radial variation, which will, in fact, occur near the end of the fuel cycle, will be 1.25 [Sterbentz 2003] times the average. This significant radial variation in the local power density will create a variation in the temperature of the helium coolant as it empties into the lower plenum. The coolant channel with the highest outlet temperature is referred to as the 'hot' channel. The concern is that the high temperature channels, which exit into the lower plenum as jets, called 'hot streaking,' will adversely affect materials in the lower plenum, the exit duct and the downstream power conversion equipment, e.g., the turbine and/or intermediate heat exchanger (IHX). The objective of the present study is to determine or bound the maximum exit temperature of the 'hot' channel coolant.

The scope of the present investigation involves the study of a single coolant channel with specified thermal and flow boundary conditions. For the thermal boundary condition, the heat flux is specified at the wall as a function of axial distance along the channel. A heat flux that varies axially based on a sine function is multiplied by the peak radial factor for the hot channel. An average channel is also modeled with a radial factor of 1.0. Because a

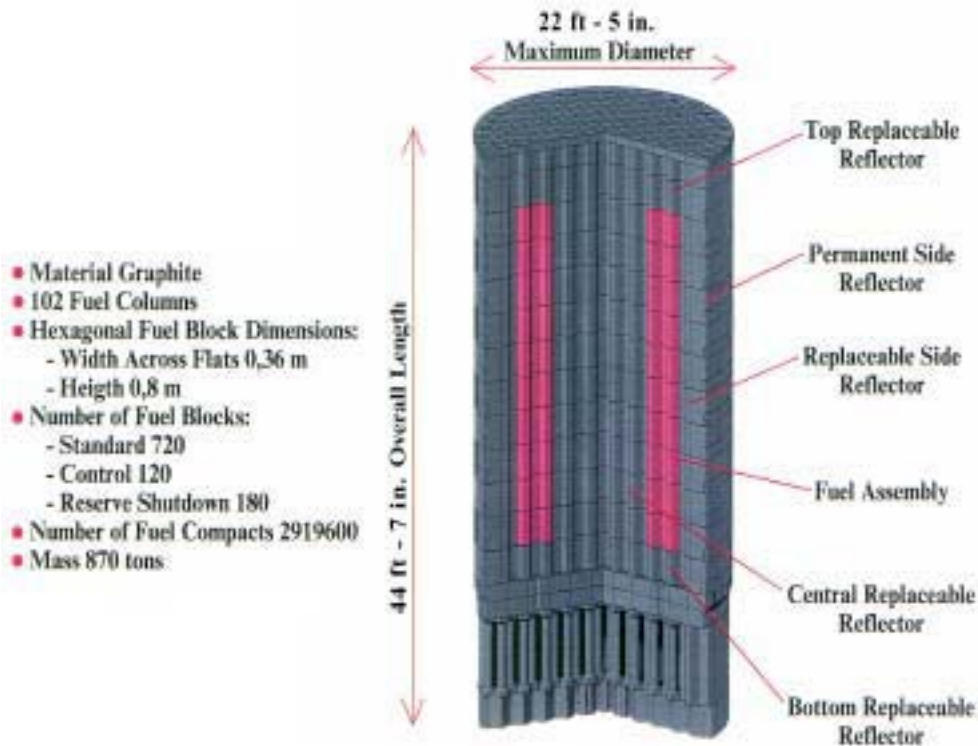


Figure 1. Layout of the core for the block NGNP point design.

heat flux specification is made for the thermal boundary condition, there is no need to specify a heat transfer coefficient to solve the problem. The flow boundary conditions are based on the specification of a pressure drop across the heated core plus the upper and lower (unfueled) reflector sections. If the flow boundary conditions could be specified using a given flow rate, then the calculation to determine the outlet temperature of the hot channel would involve a simple algebraic expression representing the heat balance. However, because of the specification of a pressure-drop flow boundary condition, the flow rates through the various channels are functions of the wall friction along each channel, which, in turn, are functions of the Reynolds number.

The Reynolds number of each of the steady (normal operation) flows in each coolant channel is basically a function of the dynamic viscosity, which increases with temperature, causing the Reynolds number to decrease as the helium is heated. However, the temperature in the hot channel is higher than that for the average channel for the same axial distance. This causes the Reynolds number to be lower for the same axial location vis-à-vis the average channel. The friction factor is higher for lower Reynolds number, so that the mass flow rate in the hot channel is lower than for the average channel. For a lower flow rate, the heating rate of the helium is increased, which further increases the temperature of the coolant in the hot channel. The accurate estimation of the flow rate of the coolant in the hot channel is dependent on an accurate representation of the wall friction (shear stress) in the channel for a flow which has a continuously changing

Reynolds number, and, hence, a continuously changing wall shear stress. Wall shear stress for both laminar and turbulent flows is well known from experiment and has been approximated by a number of algebraic expressions. The hot channel must be modeled numerically to obtain accurate wall friction, flow rate and coolant temperature estimates. Modern computational fluid dynamic (CFD) codes can provide acceptable accuracy for this problem.

2.1 Approach

The approach to compute the flow rate and temperature of the hot channel in the block NGNP is to employ the commercial CFD code Fluent [Fluent 2003]. Fluent provides a range of boundary conditions, turbulence models and the ability for the user to specify fluid properties and complex thermal boundary conditions.

Assumptions made for the flow of helium in the Fluent model include the following:

1. The heat flux into the channel is circumferentially uniform.
2. The generation of thermal energy is based on the local radial peaking factor, but must sum to the total of $600 \text{ MW}_{\text{th}}$ for all of the channels. Although each channel will experience a flow rate based on local power density, the total flow is estimated by multiplying the flow in the average channel by the ratio of the total flow area to its flow area.
3. The heat flux is a sine function from the beginning of the heated core to the end of the heated core.
4. The coolant exit temperature for each channel can be determined by the temperature at the end of the lower reflector. Although several channels combine prior to entering the lower plenum, the temperature of the combined several channels will not be greater than that of the hot channel.
5. The end of the lower reflector is a straight cylindrical channel as is the whole of the hot channel.
6. The radial power density factor translates directly into the heat flux to the local coolant channel.
7. None of the heat is removed by the bypass flow.
8. The bypass flow is 15% [Ball 2004].
9. There is no leakage of the coolant from the cooling channels.
10. The surface of the coolant channel is hydraulically smooth, such that the smooth tube data for friction factor can be used. It is believed that this is a good assumption.

11. The transition from block to block does not create a pressure drop.
12. The hot channel occurs in the coolant channel with the larger radius. In the standard block, there are 102 large channels and 6 smaller channels. The smaller channels are located in the center of the block. There are no fuel pins in the center of the block such that the temperature of the coolant in the smaller channels is assumed to be less than that of the hottest larger channels. Figure 2 illustrates the positioning of the coolant channels and fuel rods in a standard block.
13. Buoyancy effects in the coolant channel are negligible. This assumption is based on examining the results as discussed in the results section.
14. The flow is fully turbulent and does not reach a laminarizing state. This assumption is based on the results as shown in the results section.
15. The flow is incompressible. This assumption is also based on examining the results as discussed in the results section.

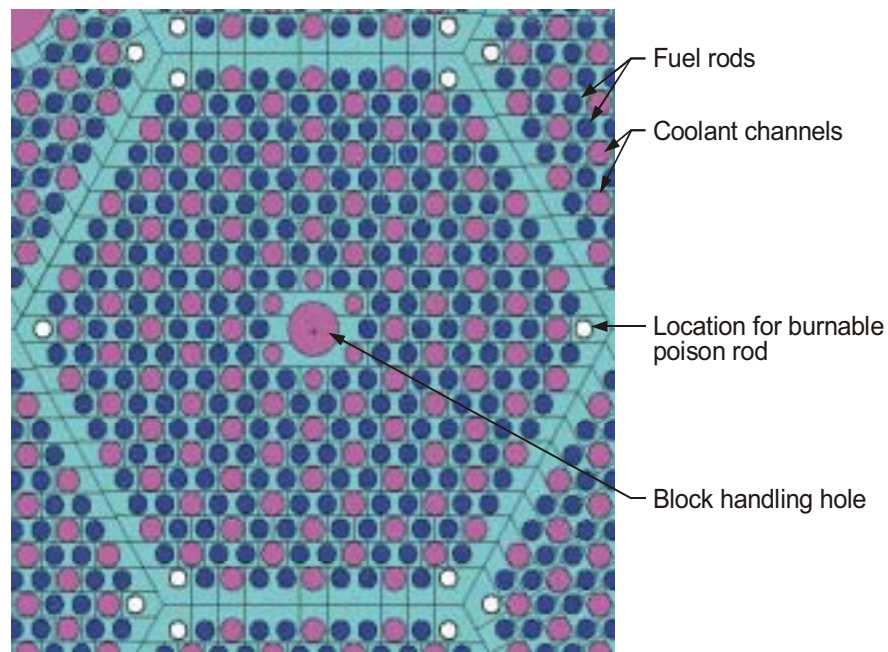


Figure 2. Layout of the coolant channels and fuel rods in a standard block of the NGNP point design. The six smaller channels surround the block handling hole.

Information on the geometry of the coolant channels and the upper and lower reflectors is taken from MacDonald et al [2003] and General Atomics [1996]. The upper and lower reflectors are 1.189 m and 1.585 m high, respectively. The heated portion of the core consists of ten 0.793 m high graphite blocks, for a total of 7.93 m height of heated core.

Combining, the total length of coolant channel is $1.189\text{ m} + 7.93\text{ m} + 1.585\text{ m} = 10.704\text{ m}$. There are coolant channels of two radii: 0.00635 m and 0.007938 m . The Fluent model is based on the channel with the larger radius.

Fluid properties needed for the CFD simulation of flow in the coolant channel include the density, the dynamic viscosity, the thermal conductivity and the specific heat at constant pressure. These properties are available from the National Institute of Standards and Technology (NIST) of the U.S. Department of Commerce¹. Helium properties are available up to 1500 K ($1226.85\text{ }^{\circ}\text{C}$). Properties for the fluid used in Fluent can be entered in one of several ways. The density can be entered in as piece-wise linear data or computed from the incompressible ideal gas law. Figure 3 illustrates the density directly from the NIST database compared to the incompressible ideal gas law for a pressure of 7.0671 MPa (1025 psi). Because the difference between the two is at most 1.5% (and at low temperatures), the ideal gas law is used in the Fluent model. Figures 4 and 5 illustrate the viscosity and thermal conductivity of helium taken from the NIST database. These data in 50 K increments have been entered into the Fluent model as piece-wise linear data in terms of Kelvins. Fluent assumes temperature is in Kelvins as a default. The fluid properties are linearly interpolated from these data. The specific heat is essentially a constant with a value of 5193 J/kg K . In Fluent, if the temperature exceeds that for the input data, the closest property value is used.

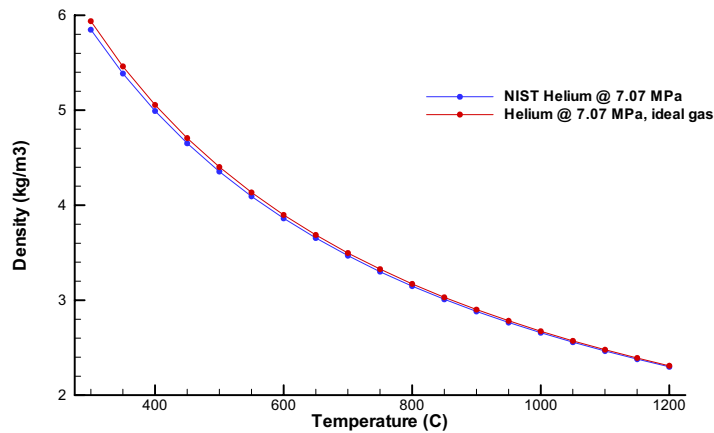


Figure 3. Helium density at 7.07 MPa (1025 psi) from the NIST database compared to results using the incompressible ideal gas law.

¹ <http://webbook.nist.gov/chemistry/fluid/>

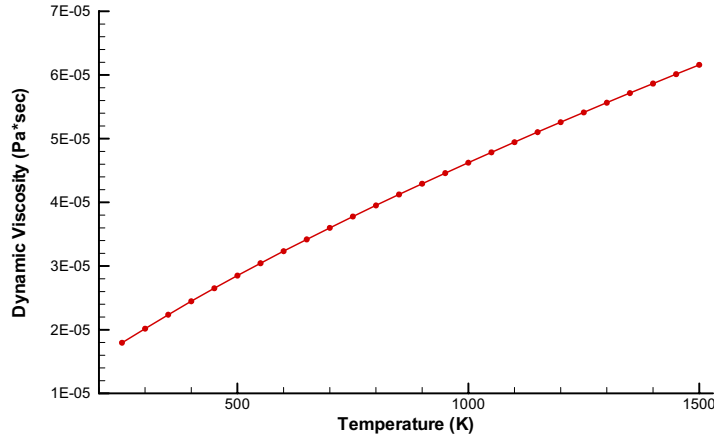


Figure 4. Dynamic viscosity for helium as taken from the NIST database.

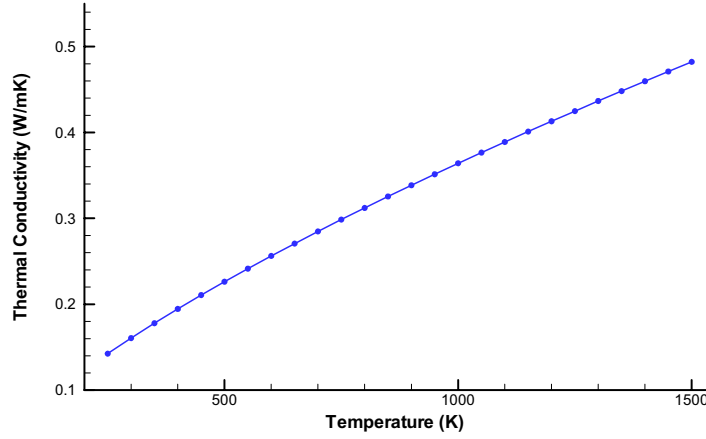


Figure 5. Thermal conductivity of helium as taken from the NIST database.

The inlet temperature is set to be 491 °C. The wall thermal boundary condition is based on a sine function. The expression derived for the wall heat flux is as follows

$$q''(x) = A_R q_{\text{con}} [1 + (A_P - 1) \sin(\pi x/L)] \quad (1)$$

where q'' is the wall heat flux in W/m^2 , x is the axial coordinate of the channel, A_R is the radial peaking factor, q_{con} is a constant necessary to specify the total heat generated in the core [when Equation (1) is axially integrated], A_P is the axial peaking factor and L is the length of the heated portion of the core (7.93 m). As can be seen in Equation (1), the radial and axial peaking factors multiply to yield a total peaking factor of $A_R A_P$ at the axial midpoint. For the average channel, the radial peaking factor is set to 1.0. Based on information obtained from Sterbentz 2003, the axial peaking factor is 1.3 and the radial peaking factor is 1.25. The latter factor actually occurs at the end of the fuel cycle [Sterbentz 2003]. Figure 6 illustrates the wall heat flux for the hot and average channels used in the Fluent model. The wall heat flux is specified using a “User-Defined Function”

in Fluent. The constant q_{con} is obtained to be $122,535.85 \text{ W/m}^2$ by integrating Equation (1) with $A_R = 1$ over the heated section to obtain:

$$Q_{\text{total}}/(A_{\text{ht,total}}/L) = q_{\text{con}}L (1 + 0.6/\pi) \quad (2)$$

where Q_{total} is the total heat generated by the core ($600 \text{ MW}_{\text{th}}$), $A_{\text{ht,total}}$ is the total heat transfer area of the coolant channels and L is the length of the heated core. If all of the channels were of the same diameter, $A_{\text{ht,total}}$ would be $N\pi R^2L$ where N is the number of channels. However, there are 9984 large channels and 642 small channels. Rather than apportion the heat to the two different sized channels by the wall areas of the channels, it is apportioned by the cross-sectional areas. This apportionment the heat based more closely on flow rate than on geometry of the channels. The true apportionment would need to be done based on the actual heat removed by each channel, which is a function of the flow rate in each channel, which in turn is a function of the local power density. Here, the heat transfer area is approximated as:

$$A_{\text{ht,total}} = 9984 (\pi D_{\text{large}}L) + 642 (\pi D_{\text{large}}L) (\pi R_{\text{small}}^2L)/(\pi R_{\text{large}}^2L) = 4111.32 \text{ m}^2$$

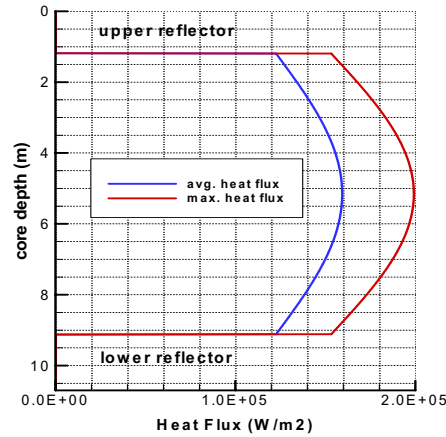


Figure 6. Wall heat flux as a function of axial depth for the hot and average channels.

The boundary conditions for the flow equations are set as zero velocity on the walls (no slip) and specified pressures at the inlet and outlet. With these momentum boundary conditions, the flow rate of helium through the channel is calculated by Fluent. It is primarily a function of the wall friction, which depends on the flow Reynolds number ($Re_D = \rho DV/\mu$), where ρ is the density, D is the channel diameter, V is the bulk velocity and μ is the dynamic viscosity. The mass flow rate is given as $\dot{m} = \rho AV$, where A is the cross-sectional area of the channel. Combining these two expressions, and $A = \pi D^2/4$, the Reynolds number can be rewritten as $Re = 4 \dot{m}/(\pi D \mu)$. Recognizing that the only parameter that varies in this expression for a particular channel is the viscosity μ , we see that the Reynolds number is a function of the viscosity only, which increases with increasing temperature (Figure 4). Because the temperature will depend on the local power density, the Reynolds number will be different for different channels for a given axial depth in the core. Finally, because the wall friction is a function of Reynolds number, the flow rate will depend on the axial bulk temperature distribution of each

channel. For hotter channels, the viscosity is higher making Re lower. The friction factor is higher for lower Re, which reduces the flow rate. A reduced flow rate causes the temperature to climb even higher. The challenge for the CFD code, then, is to correctly predict the wall friction (shear stress) as a function of Re. The temperature can be determined from a simple algebraic heat balance. For flow in the coolant channel, the heat added to the helium is given by the well-known heat balance equation:

$$Q = \dot{m} c_p [T(x) - T_{in}] \quad (3)$$

where Q is the heat transferred to the fluid (W), \dot{m} is the mass flow rate (kg/sec), c_p is the specific heat (J/kg K), and T is the bulk temperature. The heat balance is computed from the inlet to some depth 'x'. The heat is transferred through the wall as the wall heat flux q'' , given by Equation (1). By integrating Equation (1) from the inlet (of the heated core) to any point 'x' beyond the inlet, a value of Q can be found. By employing Equation (3), the bulk temperature $T(x)$ at any point x can be found. Based on the bulk temperature, the temperature-dependent viscosity can be computed, and, hence, the Reynolds number. However, Equation (3) can only be employed when the mass flow \dot{m} is known. While the mass flow for the average channel is known by assumption, the hot channel \dot{m} is not known until an accurate CFD simulation has been made.

The relationship between the friction factor f (where the wall shear stress $\tau_{wall} = f \rho V^2/8$) and the Reynolds number is well known and can be found in virtually any fluids text. For the case of turbulent flow in hydro-dynamically smooth pipes, several researchers have produced expressions for the variation of f with Re. Well known expressions include those of Prandtl, Colebrook, Blasius and the one used in the ORNL Oreca code. (No reference is given in the Oreca code manual for this expression.)

These expressions are given as follows:

$$\frac{1}{\sqrt{f}} = 2 \log \left(\text{Re} \sqrt{f} - 0.8 \right) \quad \text{Prandtl [Roberson \& Crowe 1985]} \quad (4)$$

$$f = \frac{0.3164}{\text{Re}^{0.25}} \quad \text{Blasius [Fox \& McDonald 1985]} \quad (5)$$

$$\frac{1}{\sqrt{f}} = -2.0 \log \left(\frac{e/D}{3.7} + \frac{2.51}{\text{Re} \sqrt{f}} \right) \quad \text{Colebrook [Fox \& McDonald 1985]} \quad (6)$$

$$f = 0.0014 + 0.125 \text{Re}^{-0.32} \quad \text{Oreca [Ball 1976]} \quad (7)$$

Note that the Prandtl and Colebrook expressions are transcendental. In the Colebrook expression, the parameters e/D is the relative roughness. For a smooth pipe, $e = 0$.

In the case of the hot channel, it will be shown that the temperature of the helium varies by more than 100 °C from the center to the wall. An expression to correct the friction

factor for variable fluid properties is given in Schlichting & Gersten [2000]. This expression is

$$\frac{c_f}{c_{f_{c,p}}} = \left(\frac{\rho_w}{\rho_m} \right)^{m_\rho} \left(\frac{\mu_w}{\mu_m} \right)^{m_\mu} \quad (8)$$

where

$$m_\rho = \frac{1}{2} - 4.9 \sqrt{\frac{c_{f_{c,p}}}{2}}, \quad m_\mu = 4.9 \sqrt{\frac{c_{f_{c,p}}}{2}}.$$

Here, $c_{f_{c,p}}$ is the friction coefficient at constant properties, and the ‘w’ and ‘m’ subscripts refer to the wall and the bulk values, respectively. The friction coefficient c_f is equal to one-fourth of the friction factor f .

The flow in the hot channel, as will be shown below, is turbulent inasmuch as the Reynolds number is approximately in the range of $23,000 \leq \text{Re}_D \leq 55,000$ for a pressure drop across the whole core of between 4 and 6 psi. Hence, a turbulence model must be used to simulate the helium flow. The most important aspect of the turbulence model will be to correctly represent the wall friction (shear stress) along the wall. The wall shear stress will be a function of core depth because the Reynolds number varies (decreases) as the helium moves through the core. There are several turbulence models in Fluent. The turbulence model typically consists of a model for the fully turbulent flow in the core plus some treatment of the near-wall region where, due to the no-slip boundary condition, the flow becomes viscous (essentially laminar) just adjacent to the wall. The manner in which the near-wall region is treated will have a strong influence on the wall shear stress.

2.2 Numerical Considerations

The *Journal of Fluids Engineering* (JFE) published by the American Society of Mechanical Engineers (ASME) has developed criteria for numerical accuracy for numerical-related articles submitted to the journal. The criteria are referred to as ‘The Statement of Numerical Accuracy’ [ASME 1996]. For reader convenience, this statement is reproduced below:

Journal of Fluids Engineering Statement of Numerical Accuracy

1. The basic features of the method including formal truncation error of individual terms in the governing numerical equations must be described.
2. Methods must be at least second order accurate in space.

3. Inherent or artificial viscosity (or diffusivity) must be assessed and minimized.
4. Grid independence or convergence must be established.
5. When appropriate, iterative convergence must be addressed.
6. In transient calculations, phase error must be assessed and minimized.
7. The accuracy and implementation of boundary and initial conditions must be fully explained.
8. An existing code must be fully cited in easily available references.
9. Benchmark solutions may be used for validation for a specific class of problems.
10. Reliable experimental results may be used to validate a solution.

It is desired to address these criteria as closely as possible for the present simulation with the single caveat that the only validation needed to secure the accuracy of the results is for the wall shear stress (and not the distribution of mean velocity and temperature profiles of the coolant channel flow).

The CFD code used for the present study is Fluent [Fluent 2003] version 6.1.18, employing the 2D portion of the code in double precision, running on a 12-cpu Sun 4800 workstation. The numerical method used in Fluent is the finite volume method. The discretization of the terms is given in the Users' Manual, available from Fluent's Web Site. The viscous terms are discretized using a centered difference scheme (2nd order). The pressure term is evaluated using a centered scheme. The discretization scheme for the convective terms is specified by the user. The 2nd order QUICK scheme is used herein. There is no added, only inherent numerical viscosity present in the discretization scheme. This error is minimized by using a fine enough mesh. The segregated solver is used to solve the discretized equations; the default under-relaxation factors are used. The SIMPLE algorithm is used for pressure-velocity coupling. The effect of gravity is enabled.

The issue of iterative convergence was investigated by comparing Fluent results for fully developed flow in a pipe with the exact analytical solution. This flow is also called Poiseuille flow. Iterative convergence addresses the issue of how low the tolerance levels for equation residuals need to be in order to ensure that the solution is converged. The default value for convergence in Fluent is 1.0×10^{-3} . The solution for a simple flow in a pipe, after 90 diameters, was compared to the exact solution for convergence tolerances of 1.0×10^{-3} to 1.0×10^{-7} , as shown in Figures 7 and 8.

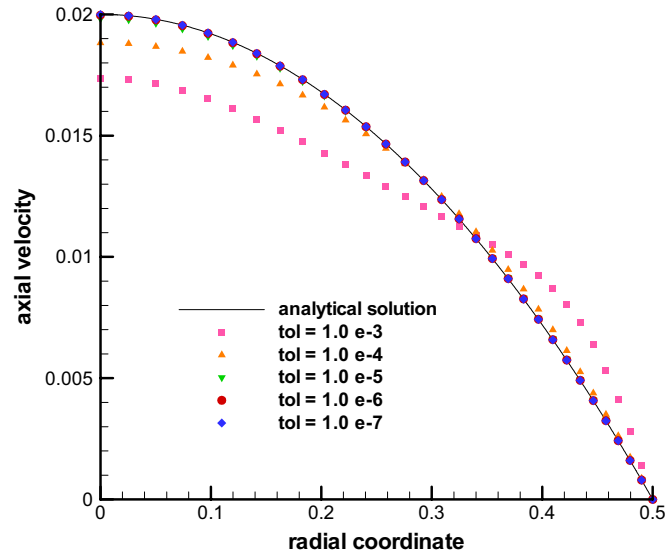


Figure 7. Solutions for a Fluent solution to Poiseuille flow in a pipe compared to the exact analytical solution for a range of convergence tolerances.

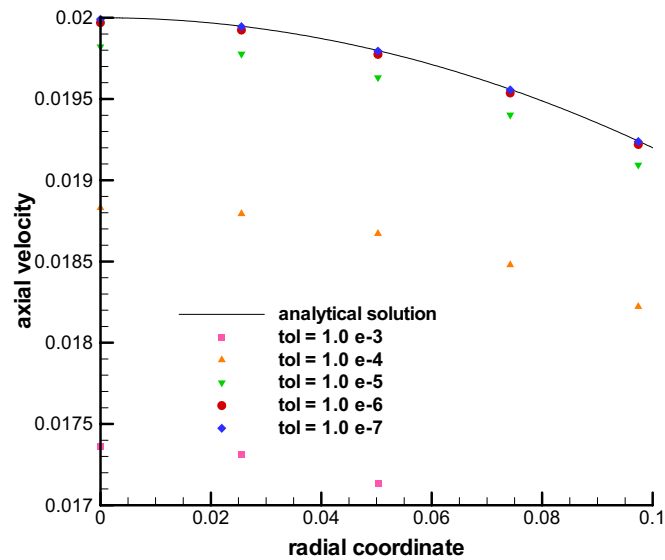


Figure 8. A magnified view of the results shown in Figure 7 near the centerline. The results for the convergence tolerance of 1.0×10^{-6} are about 0.17 % less than the exact solution at the centerline and are considered to be satisfactory.

In view of the above results, all of the simulations made were required to converge to the tolerance of 1.0×10^{-6} .

2.3 Results and Discussion

Numerical simulations of the hot and average channels were made using the commercial CFD code Fluent [2003]. Computations were made for pressure drops of 4, 5 and 6 psi for sets of hot and average channels. The flows in both hot and average channels are assumed to occur across the same pressure drop. These pressure drops include the entrance effects at the top of the core, but not the expansion and other effects of the geometry of the transition of the coolant channels just above their entrance into the lower plenum. The absence of such effects should have negligible effect on the outlet temperatures. The true pressure drops, however, will be slightly higher than for the simulations. Figure 9 illustrates several velocity profiles along the axial coordinate for the 5 psi hot channel simulation. Note that the inlet velocity is flat in the core and drops to zero at the wall. This represents a pressure drop from a flat profile and is consistent with a pressure drop at an entrance. Also note that the velocity is increasing with axial coordinate until the beginning of the lower reflector ($x = 9.119$ m) is reached.

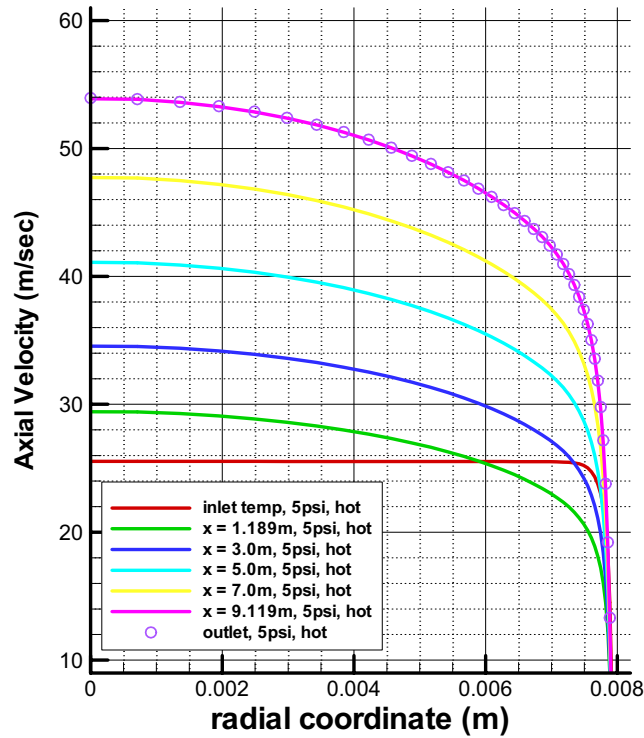


Figure 9. Axial velocity profiles for different locations along the axial coordinate for the hot channel flow with 5 psi pressure drop.

A Fluent model was created to simulate the flow in the hot and average channels. The model includes 1.189 m of unheated channel to represent the upper reflector, 7.93 m of heated channel and finally 1.585 m of unheated channel to represent the lower reflector.

The channel model is a straight tube throughout with a radius of 0.007938 m. The mesh created is a simple structured mesh with 120, 800 and 160 uniform mesh intervals in the axial direction, representing the upper reflector, the heated channel and the lower reflector, respectively. Hence, the mesh spacing in the axial direction is approximately a millimeter. In the radial direction, the mesh begins at the outer wall of the tube with a mesh element 0.000025 m wide, with a gradually increasing interval size based on 38 intervals total.

The size of the wall-adjacent interval was designed to satisfy the recommended value for the ‘enhanced wall treatment’ available in Fluent for use with the standard $k\sim\epsilon$ turbulence model. The recommendation in Fluent is that the dimensionless distance y^+ of the near-wall node from the wall be of order 1, but less than 5, where y^+ is defined as $y^+ = \rho y U_\tau / \mu$, where y is the dimensional distance and $U_\tau = \sqrt{\tau_w / \rho}$ is the friction velocity; τ_w is the wall shear stress. Figure 10 illustrates the values for y^+ for the hot channel simulations; also shown is one run for the average channel. All values of y^+ shown are within the range recommended by the Fluent manual.

The standard wall functions were initially employed with the standard $k\sim\epsilon$ turbulence model. However, the computed wall shear stress was found to vary by at least 45% from the expression of Prandtl, Equation (4). Using the ‘enhanced wall treatment’ with the standard $k\sim\epsilon$ model yielded satisfactory results for the wall shear stress.

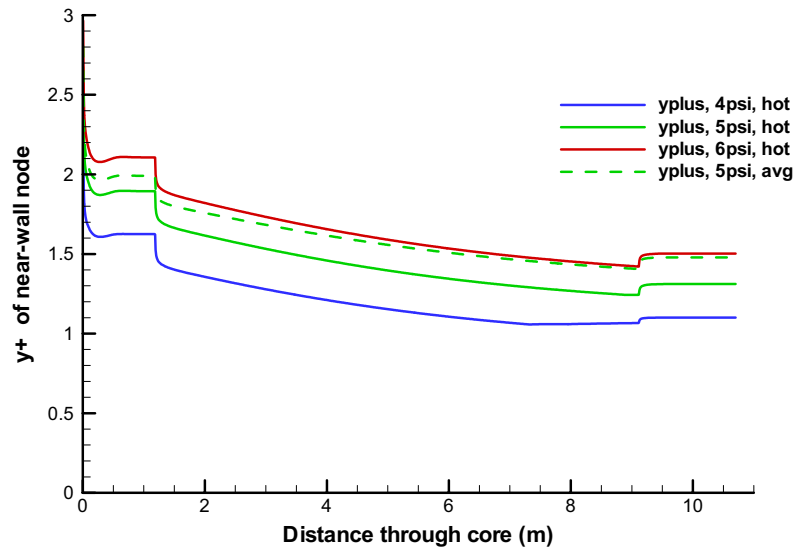


Figure 10. Axial distributions of $y^+ = \rho y U_\tau / \mu$ of the near-wall node for values of 4, 5 and 6 psi core pressure-drop for the hot channel and for 5 psi for the average channel.

The wall shear stress computed for the Fluent case of 5 psi pressure drop is compared in Figure 11 to the expressions of Blasius, Oreca, Prandtl, Prandtl corrected for variable properties, and Colebrook all given in the previous section [Equations (4-8)]. The

correction for variable properties is negligible compared to Prandtl's original expression for this flow. The maximum variations of the empirical expressions compared to the Fluent simulation are 1.1% for Blasius and Oreca and 3.5% for Prandtl, Prandtl corrected and Colebrook. Based upon these comparisons, it is concluded that the Fluent simulations are validated. It is also concluded that the near wall turbulence model is critical for this problem, while the model used for the interior is not seen to be critical.

A very fine grid calculation for the hot channel problem was made to check for grid independence. The previous grid was refined by a factor of 2 in both axial and radial directions. The distance of the near-wall node from the wall was kept the same to maintain compliance with the Fluent recommendation for the enhanced wall treatment option. The mass flow rate and outlet temperature of the hot channel calculation for the finest grid were within 0.39% of the values obtained from the next coarser grid. It is concluded that the results are grid independent.

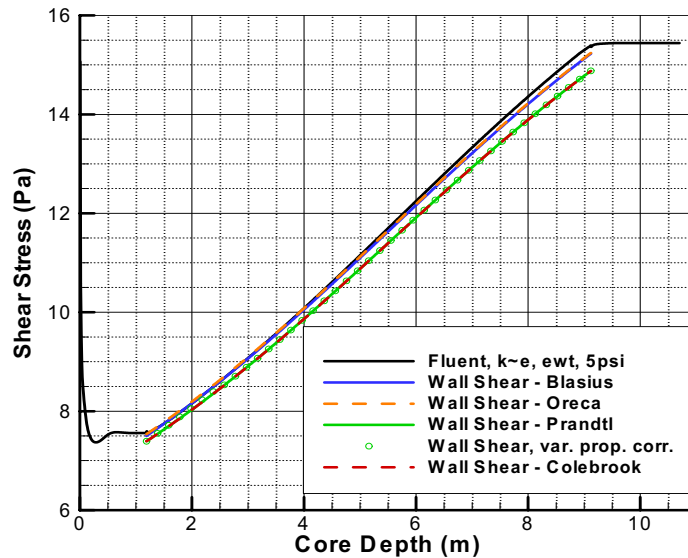


Figure 11. Fluent results for the wall shear stress for the 5 psi case compared to results using the expressions of Blasius [eq 5], the Oreca manual [eq 7], Prandtl [eq 4], Prandtl [eq 4] corrected for variable properties [Schlichting et al 2000] and Colebrook [eq 6].

The hot channel is distinguished by the fact that it assumes that the peak radial power density occurs throughout its heated length. The average channel is derived using the average power density. The flow in the hot channel is less than for the average channel inasmuch as the helium is hotter, causing an increase in viscosity, which produces a lower Reynolds number and, hence, a higher friction factor. The effect of the lower flow rate on increasing the helium temperature compounds the effect of the higher power density to further increase the temperature. Figure 12 illustrates the Reynolds numbers of the flows for the average and hot channels for the 4, 5 and 6 psi cases. Note that although the Reynolds number for all of the flows decreases with core depth, it is firmly in the turbulent flow regime ($Re_D > 5000$) for all cases.

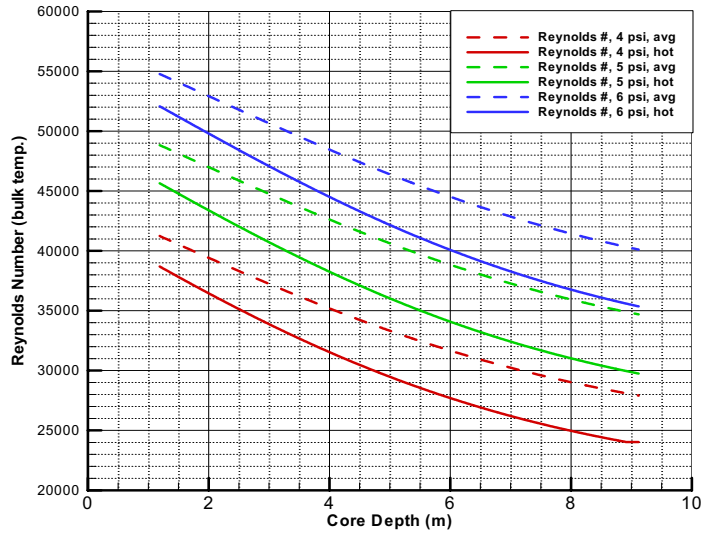


Figure 12. Distribution of Reynolds number for the average and hot channels for the cases of 4, 5 and 6 psi pressure drop across the core.

A concern for the accurate simulation of the flow in the coolant channels is the possibility of the necessity of capturing any buoyancy effects. The potential for buoyancy effects is due to the variation in temperature radially at any given axial location, which leads to fluid property variation. Figure 13 illustrates the radial temperature profiles for several axial locations for the hot channel for 5 psi pressure drop.

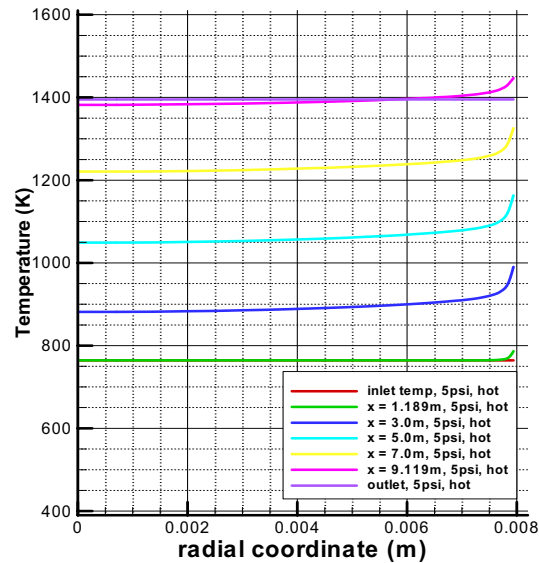


Figure 13. Radial temperature profiles for several axial locations for the hot channel at 5 psi pressure drop.

As can be seen in Figure 13, the temperature varies by as much as about 100 K across the channel cross-section. Buoyancy effects are important [Fluent 2003] if the dimensionless ratio of Grashoff number to the Reynolds number squared is of the order of one. This translates to

$$\frac{Gr}{Re^2} = \frac{g\beta D\Delta T}{V^2} \quad (9)$$

Where g is the acceleration due to gravity, β is the expansion coefficient (taken to be the inverse of the absolute temperature), V is the bulk velocity and ΔT is the temperature difference in the radial direction. This ratio is of the order of 3×10^{-5} at the most. Hence, buoyancy effects are negligible. Note from Figure 13 that the temperature profile is essentially uniform at the outlet of the channel. This is due to the fact that the turbulent heat transport has had time to fully mix the helium in the lower reflector where no additional heating is assumed and the fact that the walls of the lower reflector are assumed to be insulated.

In addition to using the QUICK scheme for convective differencing, a run with first-order upwinding was also made. The variation in the outlet temperature for the first-order case was less than 1°C. It is concluded that numerical viscosity is negligible for the grid used.

Another consideration is whether the flow is compressible. A flow is considered to become compressible for a Mach number greater than about 0.3, where the Mach number is the flow velocity divided by the local speed of sound. The speed of sound is given by

$$c = \sqrt{kRT} \quad (10)$$

where c is the speed sound, k is the ration of specific heats (c_p/c_v), $R = c_p - c_v$ and T is the absolute temperature. The maximum velocity computed, Figure 9, is about 54 m/sec, while for $T = 764$ K (491 °C) the speed of sound is about 1630 m/sec. Hence, the Mach number is well into the incompressible flow regime.

The outlet temperature of the coolant channels is a function of the pressure drop across the core because the pressure drop largely determines the coolant flow rate. The increase in the temperature of the hot channel compared to the average channel is also a function of pressure drop across the core. The outlet temperature of the average channel can actually be calculated by a simple heat balance [Equation (3)] as discussed earlier. The outlet temperature of the hot channel could also be computed from Equation (3) if the mass flow rate were known. If one were to assume (incorrectly) that the flow rate in the hot channel were the same as in the average channel, one would obtain an estimate of the hot channel outlet. Figure 14 illustrates the outlet temperatures of the hot and average coolant channel computed from Equation (3) as a function of total core flow rate assuming that the flow rate is the same as for the average channel. Also shown, are the three realistic hot channel outlet temperatures for the three cases computed with Fluent (4, 5 and 6 psi pressure drops).

The total flow is based on a scale-up of the average channel flow, assuming that the flow

is only a function of the channel cross-sectional flow area and the assumption of a 15% bypass flow. The true total core flow rate would need to be based on the actual flow rates of each coolant channel, which depend on the local power density. For the convenience of the reader, the information in Figure 14 relative to the hot channel outlet temperatures is recast as the increase from the average channel in Figure 15.

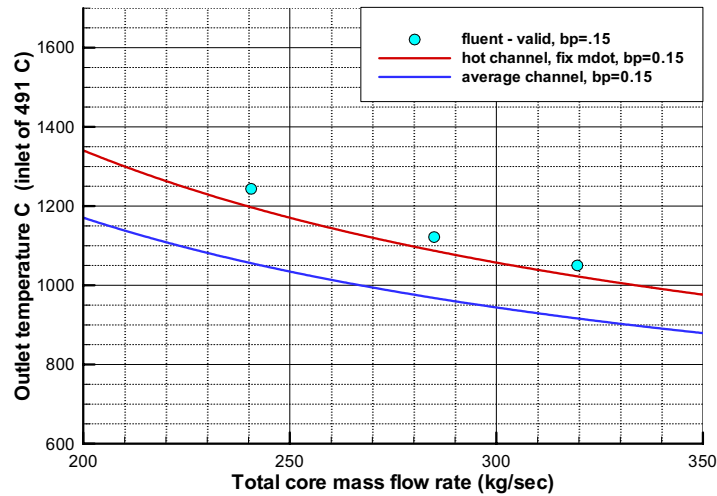


Figure 14. The outlet temperatures of the average and hot channel assuming the same flow rates plus the realistic Fluent hot channel results (symbols). The Fluent cases are for pressure drops of 4, 5 and 6 psi (left to right).

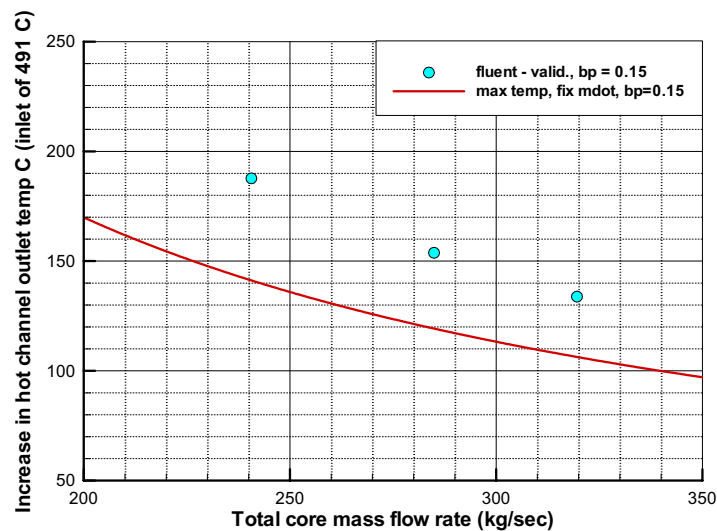


Figure 15. The increase in outlet temperature of the hot channel over the average channel assuming (incorrectly) the same flow rate (red line) and computing the actual flow rate with Fluent (symbols) for the cases of 4, 5 and 6 psi pressure drop (left to right).

In the normal operation of the actual core, the high temperatures in the regions of the graphite blocks where the maximum power densities will occur will cause there to be heat transfer to regions of lower power density. This effect is not accounted for in the Fluent model described above. However, this effect will cause a decrease in heat transfer to the hot channel. Hence, the numerical simulations will bound the maximum temperature of the hot channel outlet.

3 MIXING IN THE LOWER PLENUM

The fluid dynamics in the lower plenum of the NGNP involves the downward vertical flow of scores of hot turbulent jets of coolant exiting the core and entering into the lower plenum amid the crossflow of the coolant as it turns 90 degrees and heads to the single exit duct while negotiating the dozens of cylindrical core support columns. Figure 16 illustrates the block reactor core as well as the lower plenum and exit duct for the NGNP point design. Because the temperatures of the exiting jets of coolant will not be uniform, there are concerns of how well this complex flow will mix as it moves to the exit duct and finally to the turbine and/or the IHX. It is desired to estimate accurately the temperature of the fluid as it impinges on the lower plenum materials to address materials concerns. It is also desired to know the temperature variation of the flow as it enters the turbine as the performance of the turbine is dependent on it. It is desired to be able to numerically simulate the complex flow and heat transfer in the lower plenum. It is fully expected that a sophisticated computational fluid dynamics (CFD) code will be used. However, due to the complexity of the flow, it will be necessary to validate the CFD code, particularly the turbulence model or turbulence modeling strategy. The latter may involve the use of Large Eddy Simulation (LES) and perhaps Direct Numerical Simulation (DNS) [Speziale & So 1998].

Experiments need to be designed to capture the complex physics of the lower-plenum flow to allow assessment and validation of numerical simulations. While preliminary CFD simulations are not yet validated, they can be of use in the planning of the experiments, particularly in estimating where there are regions of high and low turbulence intensity. Mixing of the coolant is related to the turbulence intensity as well as to the overall nature of the mean flow, which will involve vortices behind support columns, etc. The purpose of the present task is to provide preliminary flow calculations of the coolant in the lower plenum to examine flow patterns and turbulence intensity. The results shown should be regarded as a qualitative prediction of the lower plenum flow behavior.

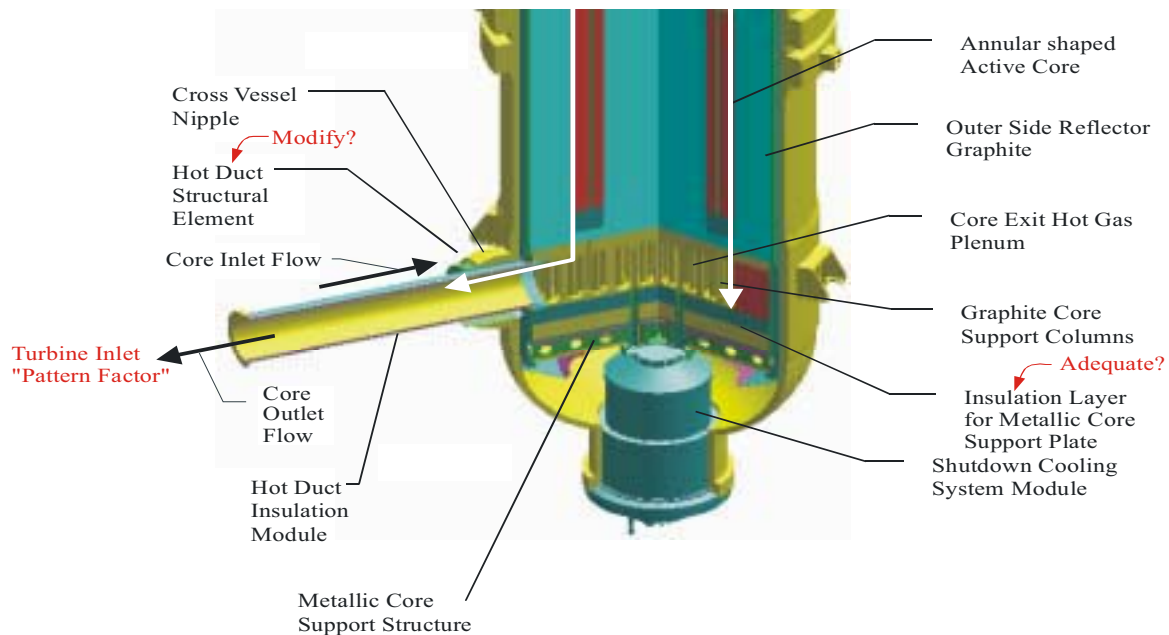


Figure 16. Layout of the block NGNP reactor design showing the core, lower plenum, the support columns in the lower plenum and the exit duct.

3.1 Objectives

The objectives of this work are twofold: (1) Perform a preliminary Fluent calculation of the mixing in the lower plenum of the General Atomics GT-MHR design at operational conditions to provide a basis for evaluating the lower plenum mixing. (2) Using the item 1 calculation, provide input to the NGNP experimentalists to guide their NGNP lower plenum test section design efforts for the Matched-Index-of-Refraction facility and heated experiments.

Although a preliminary calculation is presented in this report, it is understood that calculations will continue since the Fluent CFD code has not been validated for this application. Hence in the future additional turbulence models will be studied and used. Additional meshes may be constructed and a variety of boundary conditions will be applied in the interest of benefiting the NGNP experimentalist's design studies.

3.2 Approach and Results

The approach to performing a preliminary simulation in the lower plenum is to use a model of the lower plenum from an earlier study generated by Fluent, Inc in conjunction

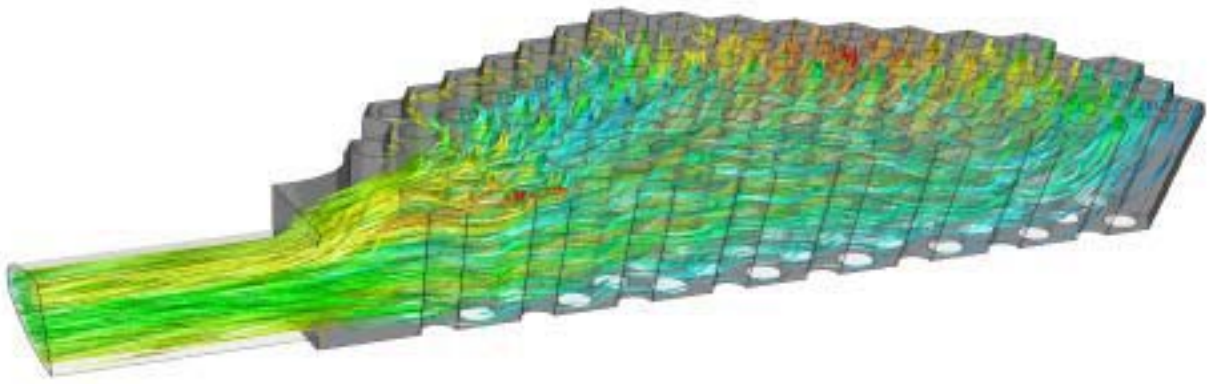


Figure 17. Fluent model of the lower plenum showing pathlines which track the flow as it enters vertically into the lower plenum, then turns toward the exit duct. (Courtesy of Fluent Inc. & General Atomics Corp.)

with General Atomics. A three-dimensional grid representing only one-half of the lower plenum was created by Fluent, Inc to represent the flow because of symmetry (Schowalter 2004). Figure 17 illustrates pathlines calculated by Fluent, showing coolant jet downflow and crossflow parallel to the exit duct.

The boundary conditions used for this study were those provided by General Atomics (Shenoy 2003). The system pressure is 7 MPa and the jet inlet temperature boundary conditions are reflected by the temperature profiles that are present at the plane of the lower plenum that coincides with the upper wall (see Figure 18).

Additional results have been obtained that illustrate the dimensionless turbulence intensity and the temperatures of the helium in the lower plenum. The present calculation differs from the original by the use of the second-order upwind momentum equation formulation and the two-equation representation of the realizable k-epsilon model. The model of the lower plenum is 2 m high. Figures 19 and 20 show the temperature and turbulence intensity for a horizontal plane located 500 mm from the bottom. Figures 21 and 22 show temperature and turbulence intensity at 1000 mm from the bottom; figures 23 and 24 illustrate the temperature and turbulence intensity at 1500 mm from the bottom.

The thermal energy is concentrated nearer the back of the lower plenum (opposite the exit duct) as the flow exits the core, but diffuses more broadly nearer to the bottom. The turbulence intensity is high as it leaves the core, but drops significantly as the bottom of the plenum is approached, especially in the regions farthest from the exit duct. The large changes in turbulence intensity as a function of location in the lower plenum presents a challenge to the NGNP experimentalists.

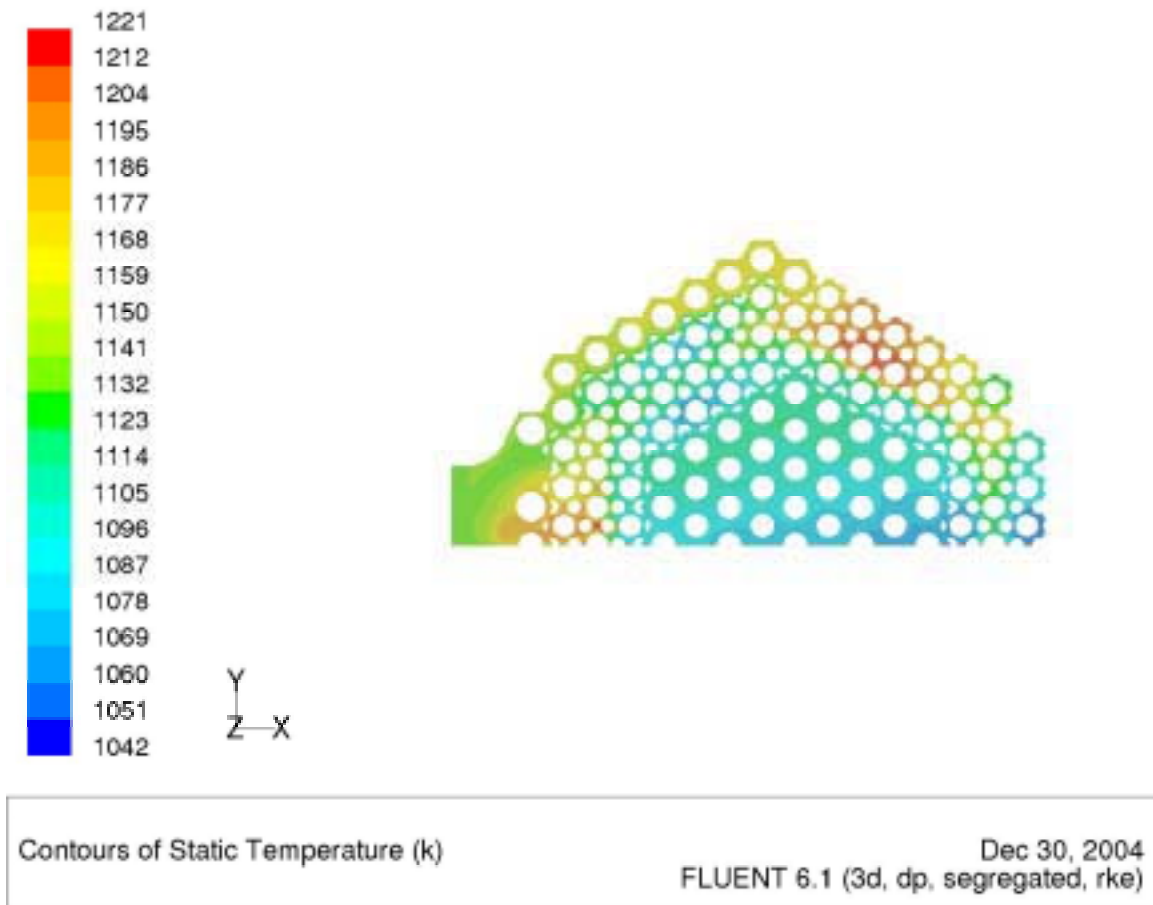


Figure 18. Static temperature contours at entrance plane of the cooling channel jets

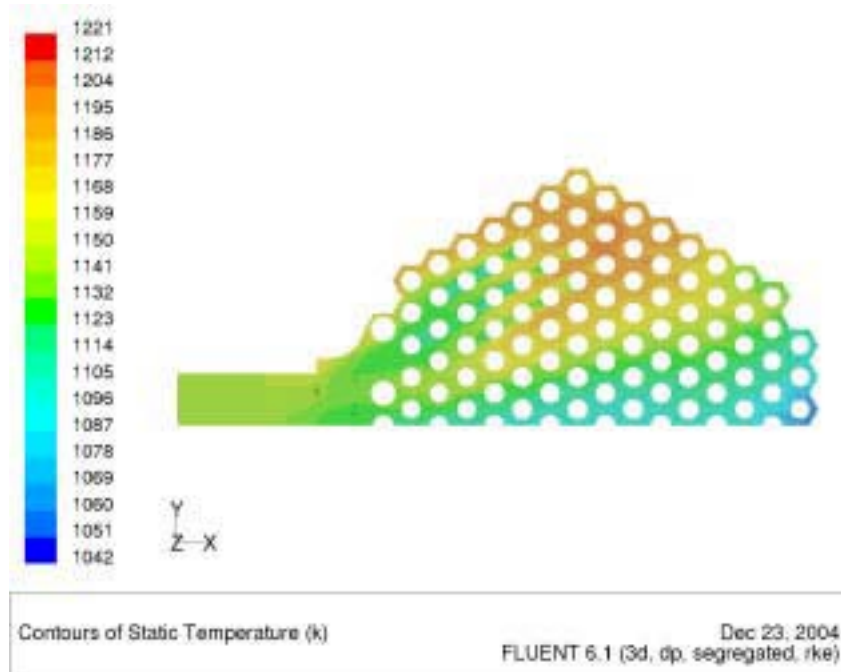


Figure 19. Temperature contours in the lower plenum 500 mm from the bottom.

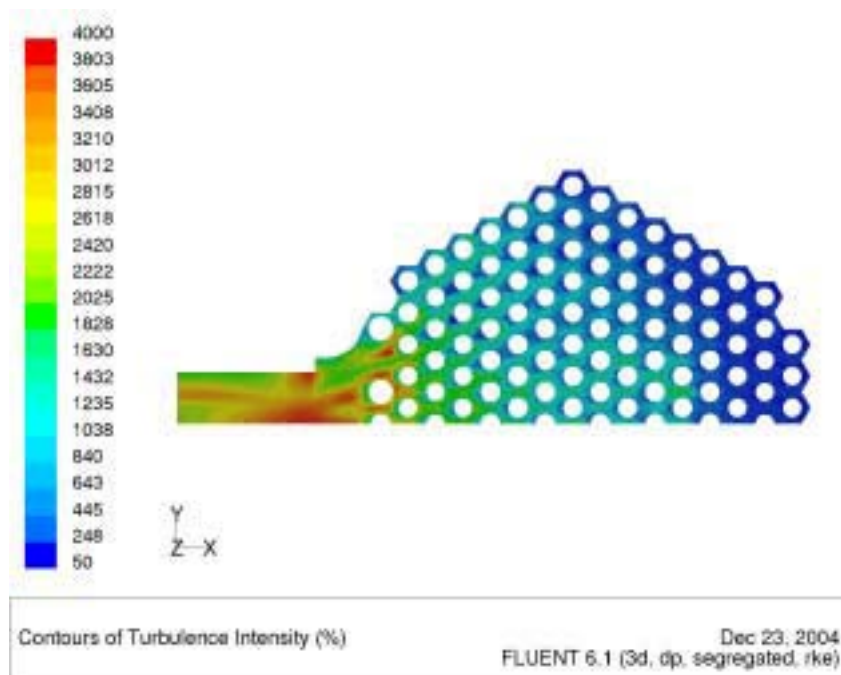


Figure 20. Contours of turbulence intensity in the lower plenum at 500 mm from the bottom.

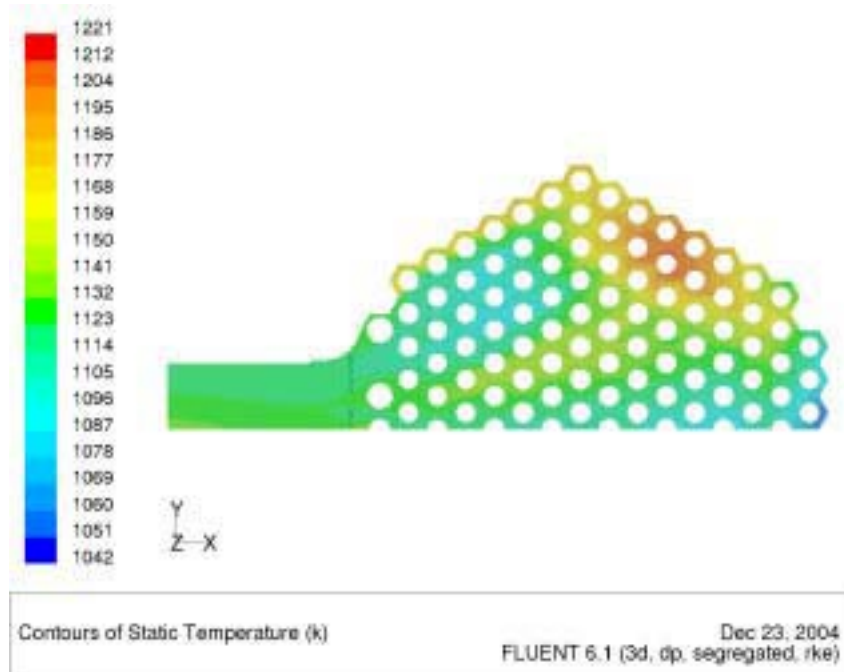


Figure 21. Temperature contours in the lower plenum at 1000 mm.

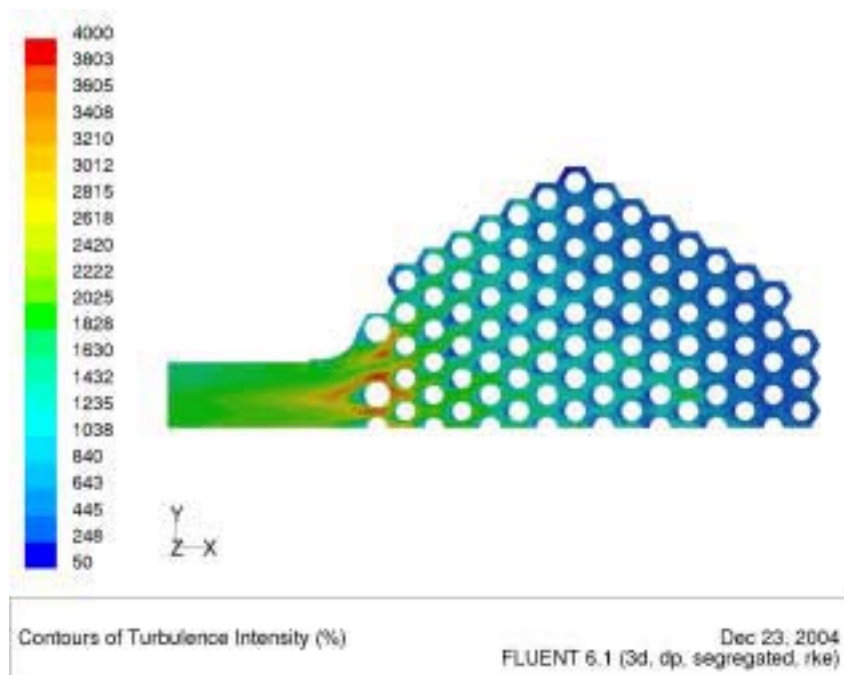


Figure 22. Contours of turbulence intensity in the lower plenum at 1000 mm.

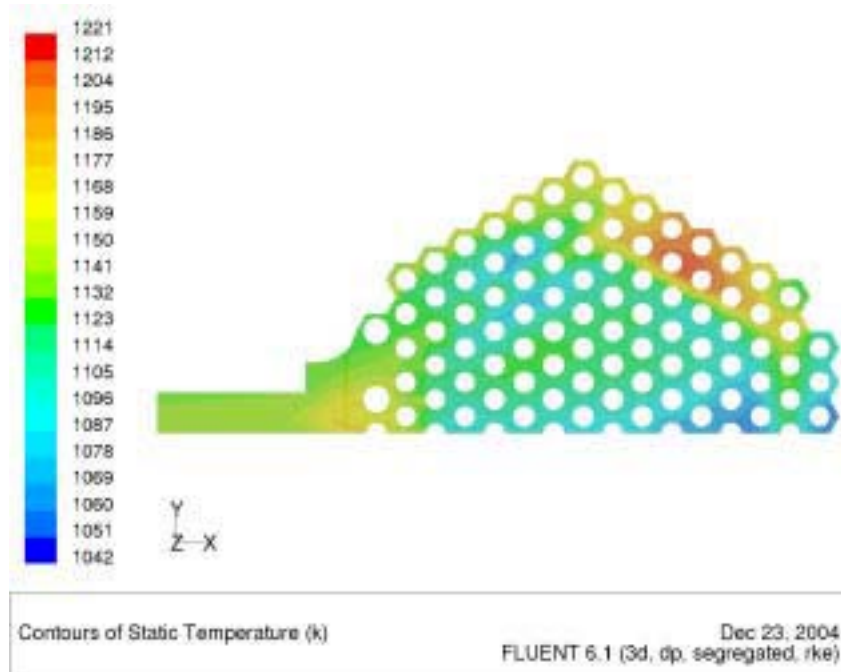


Figure 23. Temperature contours in the lower plenum at 1500 mm.

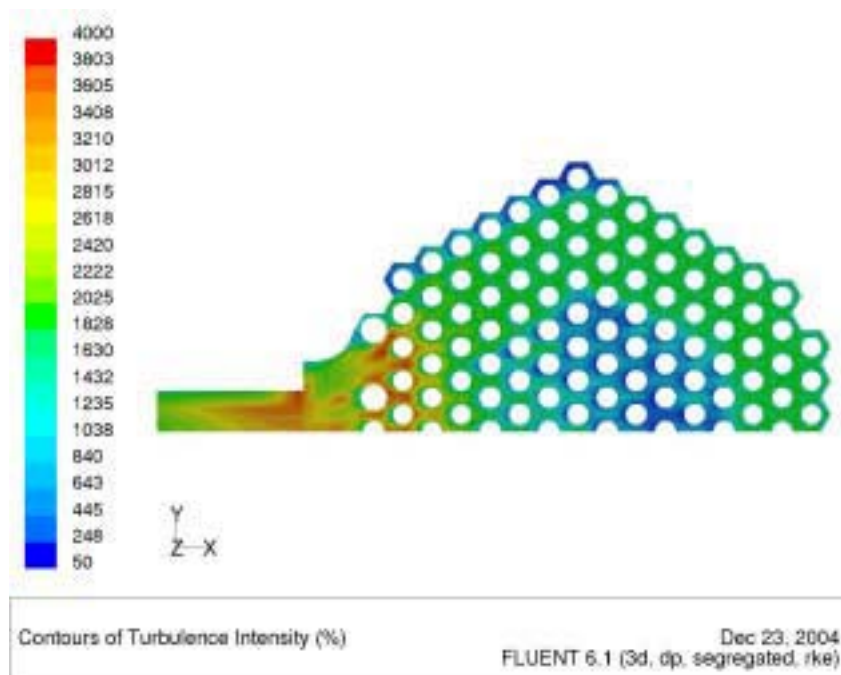


Figure 24. Contours of turbulence intensity in the lower plenum at 1500 mm.

4 CONCLUSIONS

Numerical simulations have been made of the hot and average coolant channels for the point design of a block-type NGNP to determine the increase in temperature of the hot over the average channel at the channel exits. The temperature of the average channel can be computed algebraically from a simple heat balance by assumption of the average flow rate. However, the temperature of the helium in the hot channel will be higher than for the average channel, causing the wall friction (shear stress) to increase, and the mass flow rate to decrease. This effect further increases the hot channel temperature. This effect must be calculated with a fine grid representation of the channel to allow for the change in the Reynolds number, and hence, the wall friction.

Simulations using the commercial CFD code Fluent have been performed to estimate the increase in the hot channel temperature versus that for the average channel. The criteria for the *ASME Journal of Fluids Engineering* Statement of Numerical Accuracy have been met, except possibly for the flow in the core of the channel. However, it is concluded that the correct treatment of the near wall conditions, particularly the wall shear stress, is critical to the accuracy of the desired results, while the core flow results are not. The wall shear stress must be accurate to be able to correctly determine the flow rate in the hot channel, which has a significant effect on the coolant temperature in the channel. Comparison of wall shear stress computed by Fluent to several empirically-based friction factor expressions for smooth walls shows that the Fluent calculations are within 3.5% at worst of the empirical expressions, and hence, validated.

Results show that the flow is indeed incompressible, fully turbulent for all cases studied, and that buoyancy effects are negligible. Finally, it is concluded that for the actual NGNP core, heat will be transferred away from hot regions such that the energy transferred into the hot channel will be less than calculated, making the temperatures estimated herein upper bounds on the hot channel exit temperature for the assumptions given.

With respect to mixing in the lower plenum the thermal energy is concentrated nearer the back of the lower plenum (opposite the exit duct) as the flow exits the core, but diffuses more broadly nearer to the bottom. The turbulence intensity is high as it leaves the core, but drops significantly as the bottom of the plenum is approached, especially in the regions farthest from the exit duct.

5 ACRONYMS

CFD	Computational fluid dynamics
DNS	Direct numerical simulation
DOE	U. S. Department of Energy
GT-MHR	Gas Turbine-Modular Helium Reactor

IAEA	International Atomic Energy Agency
IHX	Intermediate heat exchanger
LES	Large eddy simulation
MW _{th}	Megawatts-thermal
NGNP	Next Generation Nuclear Plant
NRC	U.S. Nuclear Regulatory Commission
R&D	Research and Development
V&V	Verification and validation

6 REFERENCES

ASME Journal of Fluids Engineering, vol. 114, June 1996, p. 427.

Ball, S. J., *ORECA-I: A Digital Computer Code for Simulating the Dynamics of HTGR Cores for Emergency Cooling Analyses*, ORNL/TM-5159, 1976.

Ball, S. J., ORNL, private communication, 2004.

Fluent, version 6.1.18, Fluent Inc., 10 Cavendish Court, Centerra Resource Park, Lebanon, NH, 03766, 2003.

Fox, R. W. and A. T. McDonald, *Introduction to Fluid Mechanics*, 3rd ed., John Wiley & Sons, New York, 1985, p. 364.

General Atomics, *Gas Turbine-Modular Helium Reactor (GT-MHR) Conceptual Design Description Report*, Report 910720, Revision 1, July 1996.

MacDonald, P. E., J. W. Sterbentz, R. L. Sant, P. D. Bayless, R. R. Schultz, H. D. Gougar, R. L. Moore, A. M. Ougouag, and W. K. Terry, *NGNP Preliminary Point Design, Results of the Initial Neutronics and Thermal-hydraulic Assessments*. Tech. Report INEEL/EXT-03-00870 Rev. 1, INEEL.

Roberson, J. A. and C. T. Crowe, *Engineering Fluid Mechanics*, 3rd ed., Houghton Mifflin Co., Boston, 1985, p. 365.

Schlichting, H. and K. Gersten, *Boundary Layer Theory*, 8th revised and enlarged edition, Springer-Verlag, Berlin, 2000, p. 552.

Schowalter, D, personal conversation with R. R. Schultz, 2004.

Shenoy, A. ltr to R. R. Schultz, "Lower Plenum Arrangement for the GT-MHR," February 4, 2003.

Speziale, C. G. and R. M. C. So, "Turbulence Modeling and Simulation," *The Handbook of Fluid Dynamics*, R. W. Johnson, ed, CRC, 1998.

Sterbentz, J. W., INEEL, private communication, 2004.